Auckland 2012
16 - 19 July
World Conference on Timber Engineering

ANALYSIS OF THE STRESS STATE OF A HALVED AND TABLED TRADITIONAL TIMBER SCARF JOINT WITH THE FINITE ELEMENT METHOD

Jose R. Aira[1], Francisco Arriaga[2], Guillermo Íñiguez-González[3], Manuel Guaita[4], Miguel Esteban[5]

ABSTRACT: The purpose of this study is to determine the stress distribution in the carpentry joint of halved and tabled scarf joint with the finite element method (FEM) and its comparison with the values obtained using the theory of Strength of Materials. The stress concentration areas where analyzed and the influence of mesh refinement was studied on the results in order to determine the mesh size that provides the stress values more consistent with the theory. In areas where stress concentration is lower, different mesh sizes show similar stress values. In areas where stress concentration occurs, the same values increase considerably with the refinement of the mesh. The results show a central symmetry of the isobar lines distribution where the centre of symmetry corresponds to the geometric centre of the joint. Comparison of normal stress levels obtained by the FEM and the classical theory shows small differences, except at points of stress concentration.

KEYWORDS: carpentry joints, halved and tabled scarf joint, FEM, stress concentration

1 INTRODUCTION

Joints are the points of transmission of forces between the members of a timber structure. In traditional joints stresses are transmitted from one piece to the other one by means of carpentry works that balance the axial and shear stresses through local compressions, tangential stresses and friction between the contact faces. The metallic elements are usually incorporated with the unique mission of keeping continuity of the contact faces [1]. The halved and tabled scarf joint consists of an end joint transmitting the tension axial force N through compression parallel to the grain located in the cross-section area of the notch b·t, and this compression is transmitted to the entire cross-section through shear stress at the surface b·l (Figure 1).

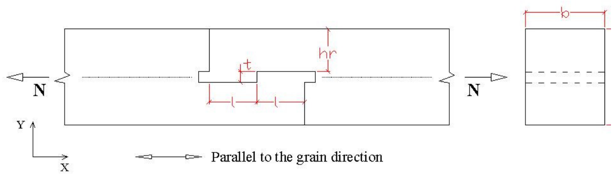

Figure. 1: *Halved and tabled scarf joint*

Moreover, the narrowing of the section in each of the pieces, resists the tensile force combined with a bending moment M caused by the eccentricity of the axial force N, producing a rotation that tends to disassemble the joint [2]. This effect can be avoided by making notches in the ends of the tabled joint or by placing metal straps. For easier installation procedure, a wedge is placed on each side to ensure tightly contact between the members (Figure 2).

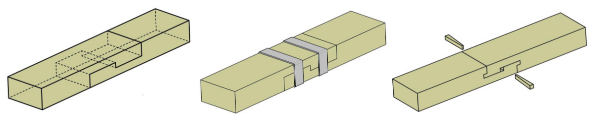

Figure 2: *Different arrangements of the halved and tabled scarf joint*

[1] Jose R. Aira. Dpto. Construcción y Vías Rurales, E.T.S. Ingenieros de Montes, Universidad Politécnica de Madrid, Ciudad Universitaria s/n, 28040 Madrid, España.
Email: joseramonaira@hotmail.com
[2] Francisco Arriaga. Dpto. Construcción y Vías Rurales, E.T.S. Ingenieros de Montes, Universidad Politécnica de Madrid, Ciudad Universitaria s/n, 28040 Madrid, España.
Email: francisco.arriaga@upm.es
[3] Guillermo Íñiguez-González. Dpto. Construcción y Vías Rurales, E.T.S. Ingenieros de Montes, Universidad Politécnica de Madrid, Ciudad Universitaria s/n, 28040 Madrid, España.
Email: guillermo.iniguez@upm.es
[4] Manuel Guaita. Dpto. Ingeniería Agroforestal, Escuela Politécnica Superior, Universidad de Santiago de Compostela, Rúa Benigno Ledo s/n, 27002 Lugo, España.
Email: m.guaita@usc.es
[5] Miguel Esteban. Dpto. Construcción y Vías Rurales, E.T.S. Ingenieros de Montes, Universidad Politécnica de Madrid, Ciudad Universitaria s/n, 28040 Madrid, España.
Email: miguel.esteban@upm.es

2 MATERIAL AND METHODS

2.1 DESCRIPTION OF THE MODEL

The geometric parameters of the studied joint are: hr = 60 mm, t = 30 mm, h = 150 mm, b = 50 mm, l = 90 mm (Figure 1). To model the joint by FEM, the left part is considered coerced on sliding supports in all nodes preventing displacement in the X axis, and a fixed support in the upper left node to prevent movement in the Y axis. The right part receives the external load of 10 kN which is uniformly distributed in the entire cross-section. A friction coefficient between contact faces of 0,467 is considered.

The three possible failure modes have been studied in the joint: a) Bending-tension failure corresponding to the reduced section of the piece subjected to tensile and bending stresses, σ_x, b) Local compression failure corresponding to the section of the notch subjected to compression stress, σ_x, and c) Shear failure corresponding to the section of the horizontal plane subjected to shear stress, τ_{yx} (Figure 3). The critical sections are studied by comparing the stress values obtained by the application of FEM to the values obtained through the formulation of the classical theory of strength of materials in order to determine the influence of mesh size on results and the coincidence of the stress distributions obtained with theoretical values.

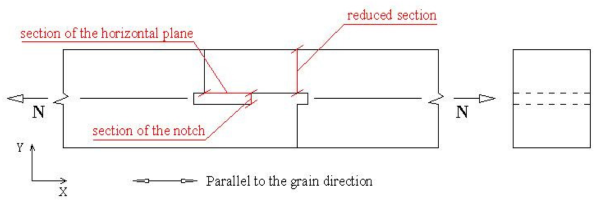

Figure 3: *Critical cross-sections*

2.2 SOFTWARE

The finite element analysis is made by a plane stress linear static study that allows the consideration of the thickness of the pieces. Wood is considered as an orthotropic material and the values of the elastic properties perpendicular to the grain are achieved by the arithmetic average in the radial and tangential directions. In order to perform the numerical simulation of the joint, each piece is modelled in the ANSYS finite element software taking the element of its internal library called PLANE42. This element is used for two dimensions modelling of solid structures and can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. The element is defined by four nodes having two degrees of freedom at each node (translations in the nodal x and y directions) and it has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities (Figure 4) [4].

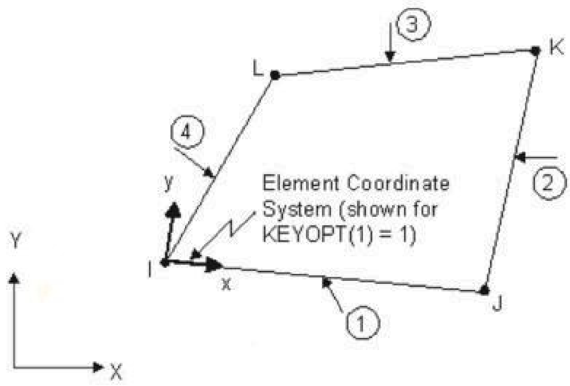

Figure 4: *PLANE 42 element*

The model includes the simulation of contact between surfaces. Groups of two different lines are defined in the contact zone established. Each of the lines belong to a different solid but having the same coordinates and geometric position in order to obtain coincidence at the nodes of each line. These lines of friction are meshed with one dimension contact elements in the direction of the lines in order to define the surface to surface contact [3]. Thus, the contact pair is set using the internal library elements called TARGET and CONTACT.

3 RESULTS

3.1 STRESSES DISTRIBUTION

The areas with stress concentration and those with lower stress are studied. To identify graphically these regions, distribution of normal stresses σ_x and distribution of tangential stresses τ_{yx} are showed from the ANSYS graphical output. Uniform mesh of size 2 mm is used (Figure 5).

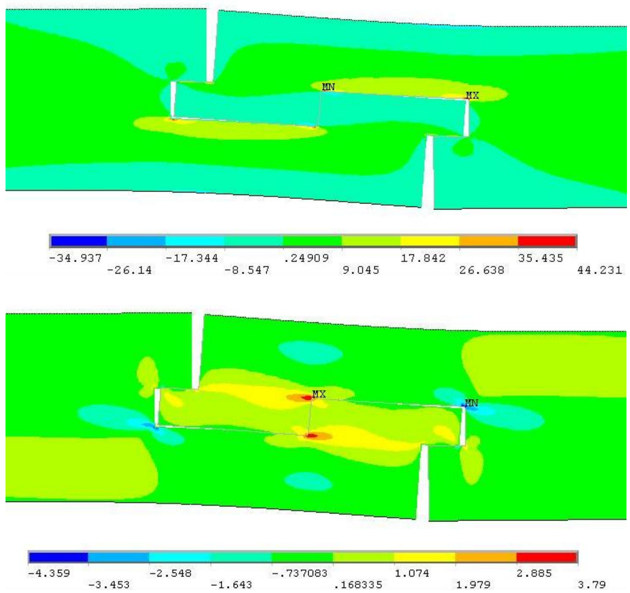

Figure 5: *Distribution of normal stress σ_x (on top) and shear stress τ_{yx} (at the bottom) in N/mm²*

The graphical output shows a central symmetry of the isobar lines distribution where the centre of symmetry corresponds to the geometric centre of the joint.

In the reduced section, subject to normal stress (bending combined with tension), there is not a point of high stress concentration except in the bottom where there is a concentration of tensile stress due to the abrupt decrease of the effective cross-section.
In the section of the notch, subject to normal stress (local compression), there are two areas of high compression stress concentration, one located at the top of the section and one in the bottom of it.
In the section of the horizontal plane, subjected to shear stress, stress values are close to zero at the end of the heel, increasing progressively to high levels of stress near the central notch.

3.2 STUDY OF THE MESH SIZE

In the finite elements model four different uniform mesh sizes are used: 10 mm, 5 mm, 2 mm and 1 mm. The computer consuming time for size 2 mm and 1 mm is too long to be functional so a type of progressive mesh is required. The progressive mesh should have the same small size in stress concentration areas and increase in size progressively when the stress concentration decreases (Figure 6). Using a progressive mesh instead of a uniform mesh, the number of nodes and finite element model is lower and consequently the number of degrees of freedom and equations to be solved by the software is also lower.

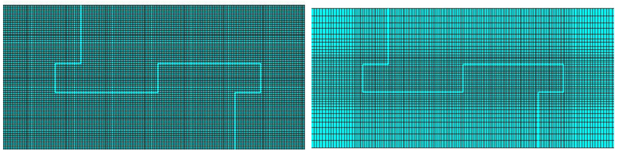

Figure 6: *Uniform mesh and progressive mesh*

To compare the accuracy of both types of mesh, the stress distribution at the critical sections is analyzed using a uniform mesh and a progressive mesh with sizes 2 mm and 1 mm.
The stress distribution for both types of mesh is identical. The stress values of the uniform mesh of size 2 mm are the same to the stress values of the progressive mesh of minimum size 2 mm. The same applies to uniform mesh of size 1 mm and the progressive mesh of minimum size 1 mm. Therefore it is possible to use a progressive mesh to provide the same precision in the results that a uniform mesh, but using a much lower computer time.
After checking the validity of the progressive mesh, to achieve greater accuracy in the results, a progressive mesh of minimum sizes 2 mm, 1 mm, 0,5 mm and 0,2 mm are used in the analysis of the critical sections.

3.3 FAILURE MODE A (BENDING COMBINED WITH TENSION)

According to the theory of Strength of Materials, the normal stress in the reduced section is obtained by the algebraic addition of the normal stress produced by the axial force N and bending moment M (Figure 7). Therefore, the normal stress σ_x is given by the expression:

$$\sigma_x = \frac{N}{b \cdot h_r} \pm \frac{M \cdot y}{I} = \frac{10000}{50 \cdot 60} \pm \frac{450000 \cdot 30}{900000} = \begin{cases} -11{,}67\, N/mm^2 (C) \\ 18{,}33\, N/mm^2 (T) \end{cases} \quad (1)$$

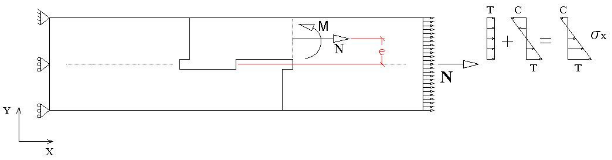

Figure 7: *Reduced section subjected to tensile force combined with a bending moment*

To compare the results obtained by FEM with theoretical values, a graph with Cartesian axes is made (Figure 8). The graph shows the stress distribution obtained by FEM along the reduced section and the theoretical stress distribution. The vertical axis represents the reduced section height in mm and the horizontal axis represents the normal stress in direction parallel to the grain σ_x in N/mm^2.

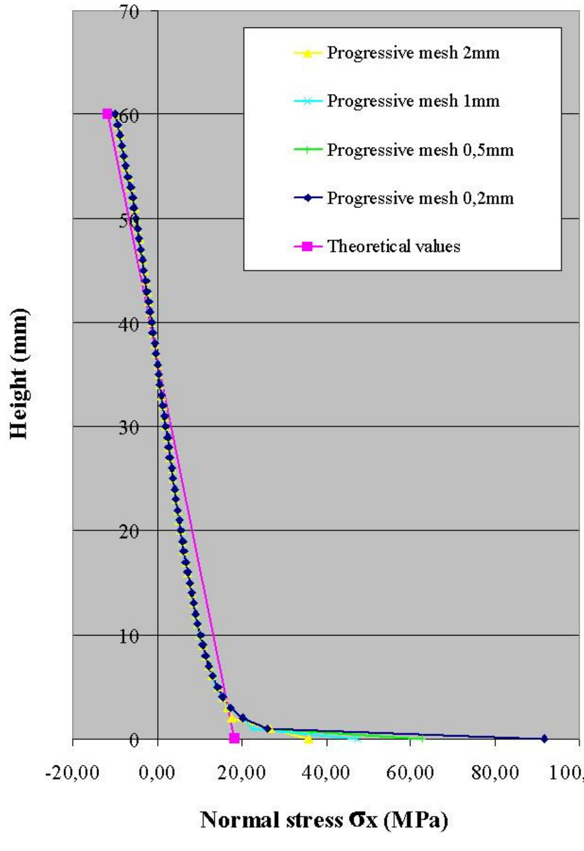

Figure 8: *Normal stress distribution in reduced section*

The stress distribution for different mesh sizes is similar between the heights of 10 mm and 60 mm in the reduced section. That is, the values of compression stress in the top of the section are very similar for different mesh sizes. The tensile stress values are also coincident except for the point where there is a stress concentration. At this point, smaller mesh size indicate higher stress values by FEM.

When the volume of stress is calculated for different mesh sizes, it is observed that the values obtained are lower than the theoretical value. When the mesh is refined, the accuracy increases and the values obtained are approaching to the theoretical value. However, with the smaller mesh size (0,2 mm), the volume of stress obtained is even greater than the theoretical value.

3.4 FAILURE MODE B (LOCAL COMPRESSION)

The normal stress σ_x in the section of the notch (Figure 9) can be obtained using the following expression:

$$\sigma_x = \frac{N}{b \cdot t} = \frac{10000}{50 \cdot 30} = 6{,}67\ N/mm^2 \tag{2}$$

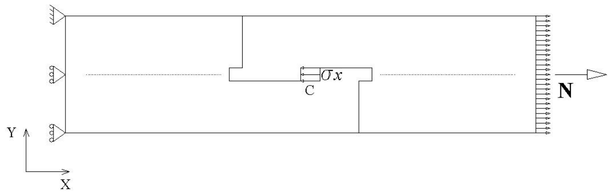

Figure 9: *Section of the notch subjected to compression stress*

In the same way, a graph with Cartesian axes is made (Figure 10) where the vertical axis represents the height of the section of the notch in mm and the horizontal axis represents the contact pressure σ_x in N/mm^2.

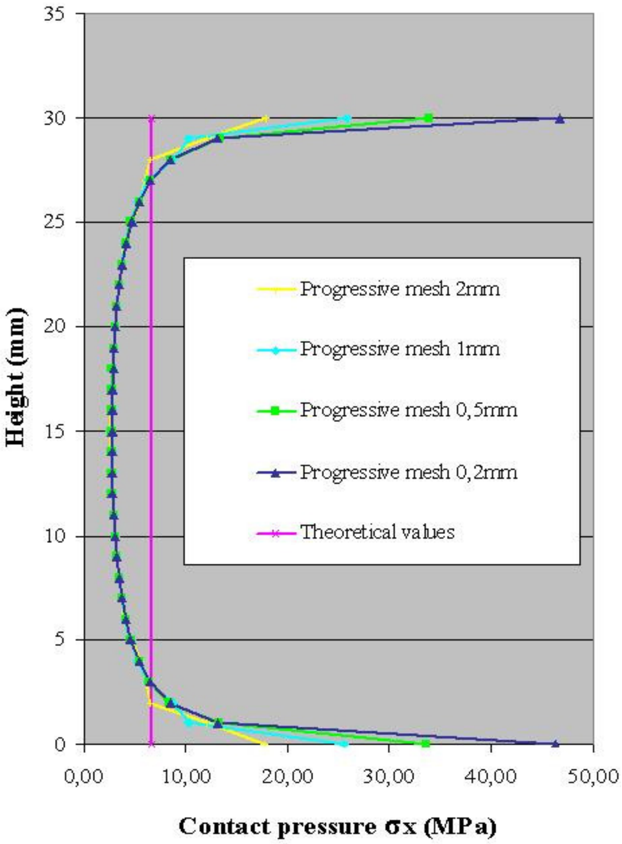

Figure 10*: Normal stress distribution in section of the notch*

The stress distribution is perfectly symmetrical. There are two areas of stress concentration, one located at the top of the section (height = 30 mm) and other located in the bottom of it (height = 0 mm). In these areas, stress values by FEM increases with decreasing mesh size.
In the rest of the section where the stress concentration is low, the stress distribution for different mesh sizes is similar and close to the theoretical value.
When the volume of stress is calculated for smaller mesh sizes, it is observed that the values obtained are lower than the theoretical value. When the mesh is refined, the accuracy increases and the values obtained are approaching to the theoretical value.

3.5 FAILURE MODE C (SHEAR)

Assuming a uniform distribution in the entire section of the horizontal plane (Figure 11), the shear stress can be obtained by the following expression:

$$\tau_{yx} = \frac{N}{b \cdot l} = \frac{10000}{50 \cdot 90} = 2{,}22\ N/mm^2 \tag{3}$$

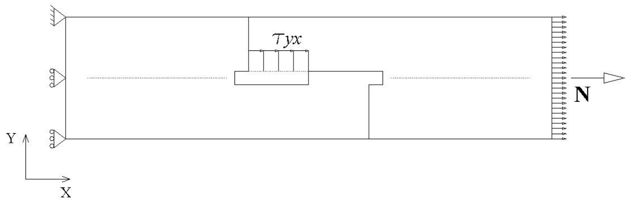

Figure 11: *Section of the horizontal plane subjected to shear stress*

As in other critical sections, a graph with Cartesian axes is made (Figure 12) where the vertical axis represents the shear stress τ_{yx} in N/mm^2 and the horizontal axis represents the length of the horizontal plane in mm.

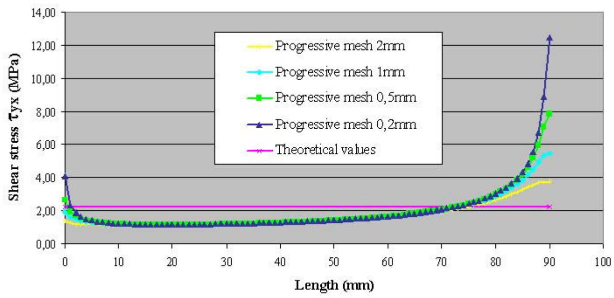

Figure 12: *Shear stress distribution in section of horizontal plane*

The stress distribution shows values that significantly increase when approaching the beginning of the heel (length = 90 mm), where stress concentration occurs. In this area, the shear stress value by FEM increases with decreasing mesh size.

4 CONCLUSIONS

Through the analysis of the stress distribution in the different sections of study for each mesh size and comparing the values obtained from the FEM and the classical theory of Strength of Materials, can be concluded:

- The results show a central symmetry of the isobar lines distribution where the centre of symmetry corresponds to the geometric centre of the joint.
- In areas where stress concentration is lower, different mesh sizes show similar stress values. In areas where stress concentration occurs, the same values increase considerably with the refinement of the mesh being necessary to refine it enough to collect the maximum stress.
- When the volume of stress is calculated for smaller mesh sizes, it is observed that the values obtained are lower than the theoretical value. When the mesh is refined, increases the accuracy and the values obtained are approaching to the theoretical value. However, an excessive reduction of the mesh size can result in volume of stress slightly higher than the theoretical value due to very high stress concentration in specific areas (Table 1).

***Table 1:** External load calculated by stress distribution in critical sections (N)*

Mesh size	Failure mode A	Failure mode B	Failure mode C
theoretical value	10.000	10.000	10.000
progressive mesh 2 mm	8.717	7.394	7.427
progressive mesh 1 mm	8.972	7.812	8.057
progressive mesh 0,5 mm	9.552	8.541	8.516
progressive mesh 0,2 mm	10.273	9.183	8.931

- Taking into account the computational resources and accuracy of the results, the correct mesh size is a progressive mesh which combines a large mesh size in areas where there is not stress concentration, with a mesh size refined enough in the stress concentration areas.
- Comparison of normal stress levels obtained by the FEM and the classical theory shows small differences except at points of stress concentration.

ACKNOWLEDGEMENT

Ministry of Science and Innovation, BIA2010-18858: "Non-destructive techniques for structural clasification of wood in new construction and rehabilitation".

REFERENCES

[1] Argüelles Álvarez R., Arriaga F., Martínez J.J.: Estructuras de Madera, Diseño y Cálculo (2ª edición). AITIM, Asociación de la Investigación Técnica de las Industrias de la Madera y Corcho Madrid (España), p.730, 2000.

[2] Arriaga F., Íñiguez G., Esteban M., Argüelles R, Fernández.Cabo J.L.: Diseño y cálculo de uniones en estructuras de madera. Maderia Sociedad Española de la Madera, p.150, 2011.

[3] Villar J.R., Guaita M., Vidal P., Arriaga F.: Analysis of the stress state at the cogging joint in timber structures. Biosystems Engineering (Biosyst. Eng), 96 (1), pp. 79-90, 2007.

[4] ANSYS: Theory Manual Version 8.1. Ansys Inc. Canonsburg, USA, 2003.